

THE CONTROL DATA "GIRAFFE"* SYSTEM FOR
INTERACTIVE GRAPHIC FINITE ELEMENT ANALYSIS

S. Park and D. M. Brandon, Jr.

Control Data Corporation

SUMMARY

This paper describes the Control Data General Purpose Interactive Graphics Application Package "GIRAFFE" (Graphical Interface for Finite Elements). The system is a general purpose interactive graphics pre/post processor for structural analysis computer programs. The system will facilitate the engineer to create, edit, or review all the structural input/output data on a graphics terminal in a time-sharing mode of operation.

INTRODUCTION

Structural Analysis problems encountered in engineering design tend to be inherently complex in nature. Establishing the structural idealization which approximates the true structure constitutes a prime problem. Also the visualization and interpretation of the solution results from the computer program is a major problem.

Traditionally, the finite element idealization is created by the engineer using structural drawings. This procedure requires a significant amount of the engineering effort due to the typical requirements of the structural computer program (element nodal coordinates, element connectivity information, etc.). The input information is usually generated as rigid format computer cards.

Depending on the complexity of the structure, this procedure results in approximately 70% of the engineer's time spent on the input data preparation, 20% for the solution interpretation and only 10% for the actual computer runs.

The General Purpose Interactive System "GIRAFFE" is developed in such a manner that the engineer can create, edit or review all the structural input/output data on the graphics terminal in a time-sharing mode of operation. Thus, GIRAFFE will enable the finite element programs to truly become more viable tools for large complex analysis problems in the structural engineering community.

Overall objectives of the GIRAFFE software are shown in Figure 1. The functional capabilities available in GIRAFFE Version 1 and Version 2 are shown in Figure 2 and Figure 3.

* This does not represent any official intent at this time to use the development name "GIRAFFE" as a product name.

OPERATING ENVIRONMENT

The operating environment of the GIRAFFE system is shown in Figure 4. The interactive graphic pre and post processing takes place in a CDC CYBER/6000 using the SCOPE/INTERCOM operating system. The terminals initially supported are Tektronix 4000 series (such as the 4010 or preferably the larger screen 4014) and the CDC CYBER GRAPHICS (777).

GIRAFFE exclusively uses the CDC GODAS (Graphically Oriented Design and Analysis System) graphic utility software. This provides for the most comprehensive graphics capabilities available such as:

- . Free format - total error recoverable dialog and graphic I/O.
- . Three-dimension interactive display creation and editing.
- . Virtual address data base and vector data base manager capability.
- . Automatic screen control (split screen usage on DVST^{*} terminals) with local (application independent) interrupt system for 2-D zooming, 3-D transformations, "hard copy cleaning", off-line plot file accumulation, etc.

The CDC GODAS uses the CDC IGS (Interactive Graphics System/basic graphic software) package for the appropriate terminal. This allows the terminals to be driven at synchronous communication rates (200 - 9600 BAUD) for faster user response.

The Finite Element Solution Phase may be run on a variety of machines using the chosen structure's application programs. The GIRAFFE system uses a "Neutral Element Library" (for finite element type definition) and produces/processes a "Neutral I/O⁺ file" for input to and output from the structure's programs.

GIRAFFE also provides for alternative sources of surface definition, in addition to direct generation, pictorially or key-in on the Graphics terminal. These alternate sources may originate from digitizers, surface scanners or interactive drafting systems.

Surface macro, which is a function of the surface definition task, provides to the user very powerful capabilities such as rotation, translation and mirror image of a basic surface. Thus, a detailed geometrically symmetric or similar structure can be easily created from a simple basic surface defined initially.

* Direct View Storage Tube

+ See the section entitled Neutral I/O Files

The data created in the Structural Data Base (SDB) by GIRAFFE can be saved permanently for the future usage. For example, if a portion of the structural component is changed at a later date, the saved tape of that SDB can be brought in to change that affected portion through surface modification or mesh modification, then perform structural analysis without recreating the whole structural data.

Under the design philosophy employing structural data base, GIRAFFE provides practically unlimited problem size capability (for example up to 30,000 elements and 35,000 nodes).

NEUTRAL ELEMENT LIBRARY

GIRAFFE provides a set of Neutral Structural Elements. The user of GIRAFFE will select one or a multiple of Neutral Elements to suit his finite element modeling requirements.

Translation from the user selected Neutral Element to the corresponding finite element of the specific application program which performs the solution phase will be handled within the application program. This gives complete freedom to the user to select the best suited program for the type of structural analysis he is performing.

The Neutral Elements available in the Version 1 of GIRAFFE are listed in the following table:

<u>Element Name</u>	<u>Description</u>
EN2TRUS	Truss element
EN2BEAM	Beam element
EN2TUBE	Tube element
EN2LSP	Linear translational spring element
EN3TSP	Linear torsional spring element
EN3MEM	Triangular membrane element
EN3PLT	Triangular plate element
EN3PM	Triangular plate element with membrane, bending and shear in effect
EN3PM1	Triangular composite element
EN4MEM	Quadrilateral membrane element
EN4PLT	Quadrilateral plate element
EN4PM	Quadrilateral plate element, with membrane, bending and transverse shear in effect
EN4PM1	Quadrilateral composite element

NEUTRAL I/O FILES

The Neutral I/O files are generated automatically. When the user completes input data preparation using GIRAFFE, he generates a "Neutral Input" file through GIRAFFE. He will subsequently submit a batch job to execute the structural application program (whichever he chooses) with the Neutral Input file as input source.

When Neutral Output file returns to a Graphics Computer it is attached as the Neutral Output file accessible to GIRAFFE. The Neutral Output file contains displacements and stresses. GIRAFFE V.1 will provide post processing activities such as deformation and stress analysis. GIRAFFE V.2 will provide deformation, stress plots and modal shape plots, and time response history plots.

HIGHLIGHTS OF THE GIRAFFE MODULES

GIRAFFE is an integrated pre/post processor to a structural application program with its own data base (see figure 5). In an interactive graphic time-sharing mode, GIRAFFE facilitates generation of the structures element modeling; i.e., element connectivities and node points data.

The objective of the GIRAFFE does not end here but is extended further. It allows preparation of all the input to a structural application program. The complete task options available in Version 1 are shown in the Master Menu listed below as displayed on terminal.

Task Options:

- 1 Terminate
- 2 Surface definition
- 3 Element mesh generation
- 4 Element mesh geometry check
- 5 Structure's property definition
- 6 Restraint definition
- 7 External load definition
- 8 Bandwidth optimization
- 9 Write neutral input file
- 10 Read neutral output file
- 11 Deformation analysis
- 12 Stress analysis
- 13 Status

Enter Task Number

The rest of this section shows only the main menu list for each task in the GIRAFFE V.1.

(1) SURFACE DEFINITION

This is the basic surface (outline or part definition) task. This task allows the user to create a basic surface (whether the source is direct CRT or batch input) and modify or delete surface data from the SDB.

Surface Definition Options

- 1 Return
- 2 CRT input
- 3 Batch input/modify
- 4 Surface macro

Enter task number

(2) ELEMENT MESH GENERATION

This task generates a finite element model either completely automatically or one-by-one manually. Also, elements can be modified. Finite element types are defined in this phase:

Element Mesh Generation:

- 1 Return
- 2 Select other structure and/or block
- 3 Define sub-block for mesh generation
- 4 Auto mesh
- 5 Manual mesh - define/edit
- 6 Delete mesh generated

Enter Task Number

(3) ELEMENT MESH GEOMETRY CHECK

Under this option the user can perform the following element mesh geometry check:

Element Mesh Geometry Check:

- 1 Return
- 2 Select other structure and/or block
- 3 Show block mesh boundary
- 4 Quadrilateral planarity check
- 5 Element normal check
- 6 Define or modify element normal

Enter Task Number

(4) STRUCTURE'S PROPERTY DEFINITION

Under this option the user can define a structure's material properties and physical properties; also, output stress reference angle. The user can request material and physical property ID numbers to be displayed on the element mesh.

Property Definition:

- 1 Return
- 2 Select other structure/block
- 3 Define material properties
- 4 Define physical properties
- 5 Display/cancel mode numbers
- 6 Display/cancel element numbers
- 7 Display/cancel material property ID
- 8 Display/cancel physical property ID

Enter Task Number

(5) RESTRAINT DEFINITION

Under user's direction, this task restrains freedoms of the nodes. The user can also define enforced boundary node displacements of the nodes. The main task menu is listed below as displayed on terminal.

Restraint Generation:

- 1 Return
- 2 Other structure/block
- 3 Define restraints
- 4 Display restrained nodes
- 5 Define boundary displacements
- 6 Display displaced nodes
- 7 Display/cancel node number
- 8 Display/cancel element numbers

Enter Task Number

(6) EXTERNAL LOAD DEFINITION

This task assigns external loads to the nodes or elements upon user's commands. GIRAFFE V.1 load options are:

- . Concentrated load(s)
- . Pressure load(s)
- . Gravitation load
- . Beam load(s)

Multiple load case definition is allowed. The main menu for this task is listed below:

Load Options:

- 1 Return
- 2 Other structure/block
- 3 Load input
- 4 Display load
- 5 Display/cancel node numbers
- 6 Display/cancel element numbers
- 7 Change Load Case

Enter Task Number

(7) BANDWIDTH OPTIMIZATION

This task selection allows the user to resequence node numbers to reduce matrix (stiffness) bandwidth for solution efficiency. This is an optional task. The user who wants to utilize built-in application bandwidth optimizer may not exercise this option. The main menu for this task is:

Bandwidth Optimization:

- 1 Return
- 2 Select other structure

- 3 Optimize bandwidth
- 4 Display/cancel node number

Enter Task Number

(8) WRITE NEUTRAL INPUT FILE

This is the last task of the GIRAFFE pre-processing phase. When the user completes the generation of all the necessary input data for structural analysis, he will select this option to generate neutral input file. This file will contain all the necessary and compatible data for the structural application programs. Upon exit from GIRAFFE he submits a batch application job with the neutral input file as the input source.

GIRAFFE dialogs in this task are:

- . Which structure's neutral input file is to be generated if there are multiple structures in SDB.
- . Build load set from the predefined load case in the load module.

(9) READ NEUTRAL OUTPUT FILE

This task reads the application program solution data saved in neutral output file. The user can subsequently process stress and displacement data in the interactive graphics mode.

GIRAFFE dialogs in this task include:

- . Enter neutral output file set number corresponding to previously created neutral input file set.
- . Results analysis for a single load set.
- . Results analysis via load set superposition.

(10) DEFORMATION ANALYSIS

Under this task the user can generate deformation plots superimposed with original structure with boundary lines only, or he can create entire mesh deformation superimposed with original mesh. The latter option should be used for detail deformation study associated with a specific block. The main menu of this module is shown below.

Deformation Analysis:

- 1 Return
- 2 Other block(s)
- 3 Limit deflection node display
- 4 Boundary deformation plot
- 5 Entire element deformation plot
- 6 Display/cancel node numbers

Enter task number

(11) STRESS ANALYSIS

Under this task control, the user can investigate stress distribution of the structure for the applied load condition. The main menu of this module is shown below.

Stress Analysis:

- 1 Return
- 2 Other block(s)
- 3 Beam-type element stress
- 4 Element stress display
- 5 Stress contour plot
- 6 Display/cancel node number(s)
- 7 Display/cancel element number(s)

Enter Task Number

(12) STATUS

This task provides the user some essential status summary of the GIRRAFE usage plus structural data base interpretive dump.

Status Summary:

Time: _____
Elapsed clock time this session: _____
Total elapsed clock time: _____
CPU time this session: _____
Total elapsed CPU time: _____
Page size: _____
Number of page transfers⁺ this session: _____
Total number of page transfers: _____
Amount of disk space allocated: _____
Number of structure: _____
 Structure: _____
 Name: _____
 Number of blocks: _____
 .
 .

⁺ This is transfer between central memory and disk

The interpretive SDB dump options are listed below:

Enter Option Number:

- 1 Return
- 2 Structure list
- 3 Block list
- 4 Surface line lists
- 5 Surface polygon list
- 6 Mesh list
- 7 Boundary line list
- 8 Elements list
- 9 Node point list
- 10 User node number and resequenced node number
- 11 Material and stress angle list
- 12 Physical property name and value list
- 13 Material property name and value list
- 14 Load lists
- 15 Boundary displacement lists
- 16 Node displacements
- 17 Output reaction force lists
- 18 Stress list
- 19 Element library list

STRUCTURAL DATA BASE

The Structural Data Base can be regarded as an unlimited memory where data is organized in lists. The main purposes of the SDB are to provide a permanent record of Structural Data and to provide communication between functional modules and for system save/restart.

The data base manager system allows the user to create, access, and modify lists on a random basis. The main purpose of the SDB Manager is to free the functional module from all I/O concern and associated code.

The data base requirements of three-dimensional analytic geometry software and other structural tasks are quite complex:

- 1 Large amounts of data must be manipulated.
- 2 A variety of algorithms and procedures are used which require access to data in both a sequential and random fashion.
- 3 A variety of different data "collections" are involved.
- 4 Maximum efficiency is mandatory to provide tolerable response time to the user at the terminal.

The GIRAFFE data base is composed of two files: one for real numbers and the other typically for integer numbers. Each of these two files logically consists of a list of sequential binary data words. The two files

physically consist of a collection of "pages" (Data Records). Data collections or "subfiles" are indicated by starting address pointers, as shown in Figure 6. The data base routines make the computer appear to be an efficient "virtual memory" device, handling all mapping, paging, and I/O.

A number of real pages and integer pages will be held in central memory for each of the two files. Pages are automatically "swapped" into central memory as required (or demanded). Pages are swapped out of central memory according to a "least recently used procedure".

The Data Base Manager has the additional capability to perform "macro" operations as well as the I/O of individual words. These macro routines take the overhead (of mapping, swapping, etc.) involved and reduce it by the page size (or length of the macro operation if less than the page size). The macro routines also perform the necessary arithmetic and data movement functions simultaneously with the I/O, thus considerably reducing the programming effort and storage required. Figure 7 shows some macro capabilities which are extensively used in analytical geometry and structural engineering calculations. This operation is conceptually shown in Figure 8.

The overall system is optimized (trading off central memory storage with central memory residency interval (CP & I/O time) by simply manipulating pages sizes (real and integer).

With one Data Base and Data Base Manager, development, documentation, and continuation costs are reduced and flexibility for modifications and additions is increased.

GIRAFFE APPLICATION EXAMPLE

Some of the major structural tasks are illustrated in Figures 9 through 25 for a simple 3D plate problem.

OVERALL OBJECTIVES

- . An innovative, powerful, flexible design/analysis tool for the structural engineering community.
- . Cost/effective software system implementation designed for efficient use in an interactive time-sharing environment.
- . Terminal independence (i.e., structural data base can be accessed by any GODAS-supported terminals either low cost DVST or high performance CDC 777 refresh terminal or future undefined high performance devices).
- . Large reduction in user's design and redesign cycle time.
- . Minimum user capital outlay for Graphics terminal.

Figure 1.- Overall objectives.

MAJOR FUNCTIONAL CAPABILITIES

SYSTEM

No practical limit on problem size
Ability to continue interrupted work
On-line real time hard copy plot
GERBER, CALCOMP, Houston Instrument, SC4020 plots can be on/off-line processed

DISPLAY UTILITY

Windowing
Zooming
Basic transformations (rotation, etc.)
Other transformations (perspective, etc.)
Hidden line removal on surface definition

PREPROCESSING

Basic surface definition
Model geometry check; boundaries, normals and planarity of quadrilateral elements
Node and/or element number display and delete
Element generation and edit can be in any sequence
Property specification
Material property ID display
Physical property ID display
Automatic 2-D mesh generators
Automatic 3-D mesh generators (plate and shell problems)
Surface definition input (interactive)
External forms of surface definition input
Manual mesh generator/editor
Bandwidth optimizer (Gibbs-Poole-Stockmeyer method)
Load specification
Display and edit load
Restraint generator
Display and edit restraint

INTERFACE TO SOLUTION PHASE

I/O interface to various analysis packages: neutral I/O scheme
Neutral element library (classical elements)

POSTPROCESSING

Display of stress values including critical limit
Display of deformations including critical limit

Figure 2.- Major functional capabilities (Version I).

MAJOR FUNCTIONAL CAPABILITIES

SYSTEM

No practical limit on problem size
Ability to continue interrupted work
Can merge data base to form a new large data base (SDB)*
Can extract one data base from other
On-line real time hard copy plot (on Tektronix 4000 series)
Gerber, CALCOMP, Houston instrument, SC4020 plots can be on/off-line processed

DISPLAY UTILITY

Windowing
Zooming
Basic transformations (rotation, etc.)
Hidden line removal
Other transformations (perspective, etc.)

PREPROCESSING

Basic surface definition
Model geometry check; boundaries, normals, and planarity of quadrilateral elements
Node and/or element number display and delete
Element generation and edit can be in any sequence
Property specification
Material property ID display
Physical property ID display
Automatic 2-D mesh generators
Automatic 3-D mesh generators (plate and shell problems)
Automatic 3-D solid mesh generators
Surface definition input (interactive)
External forms of surface definition input
Manual mesh generator/editor
Bandwidth optimizer (Gibbs-Poole-Stockmeyer method)
Curved surface mesh generation (isoparametric elements)
Load specification
Display and edit load
Restraint generator
Display and edit restraint
Temperature specification
Display and edit temperature model

INTERFACE TO SOLUTION PHASE

I/O interface to various analysis packages: Neutral I/O scheme
Neutral element library (classical elements)
Neutral element library (isoparametric, solid and special elements)

POSTPROCESSING

Display of stress values including critical limit
Stress contour plot
Display of deformations including critical limit
Boundary deformation W/W overlapping original boundary
Load case superposition to produce composite deflection plot
Modal shape plot
Structural time response history plot

* Structural Data Base

Figure 3.- Major functional capabilities
(Version II).

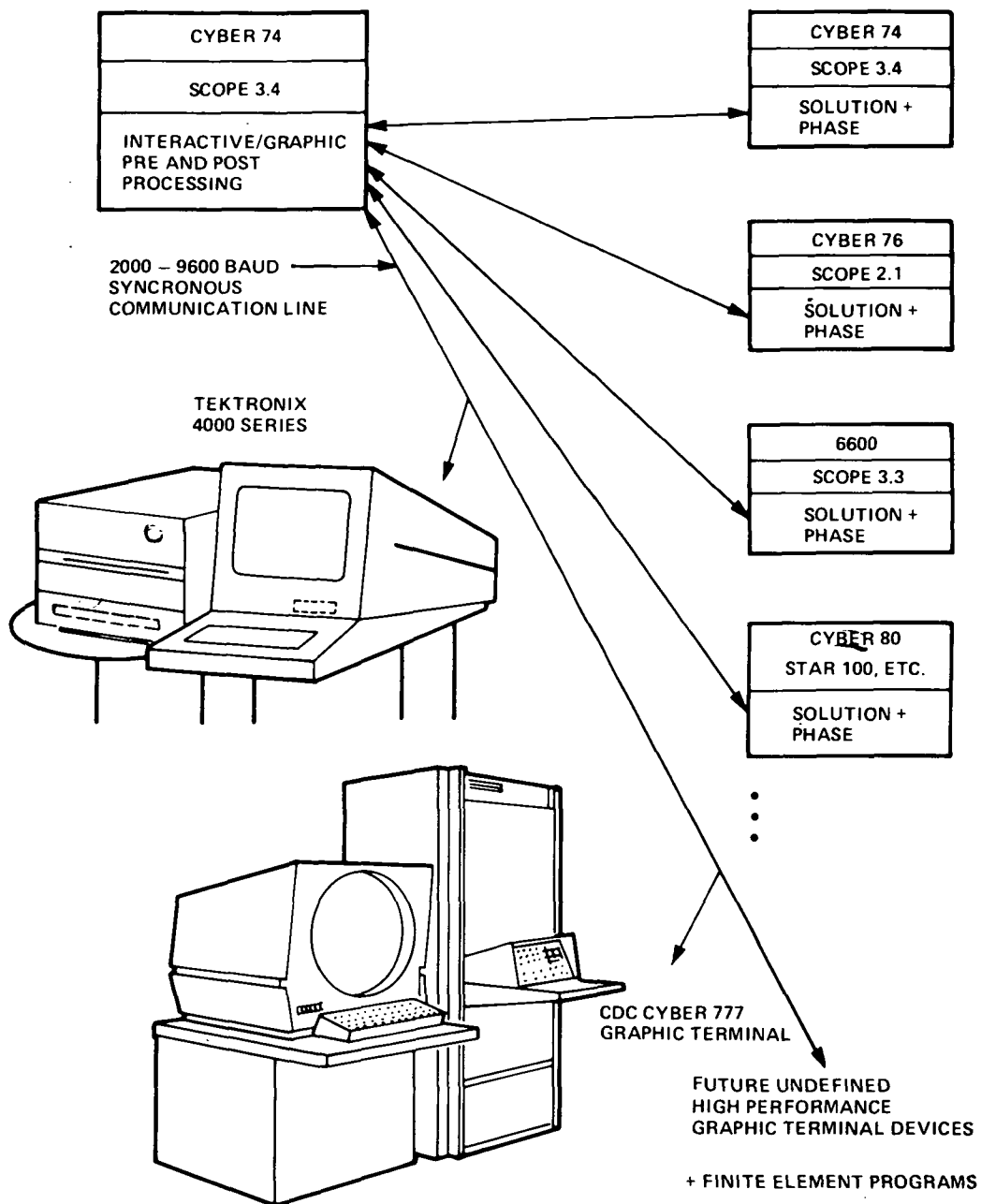


Figure 4.- Operating environment.

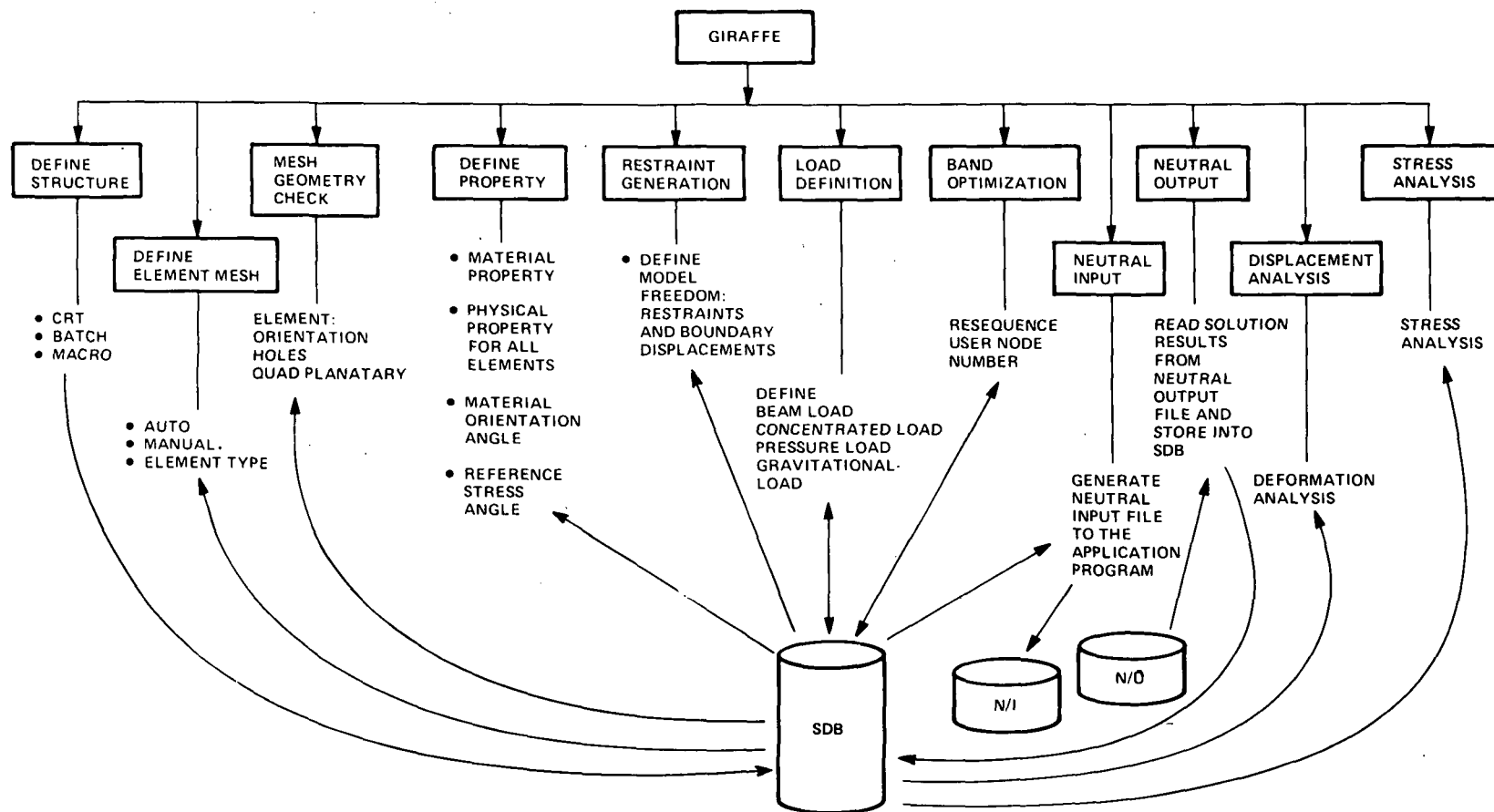


Figure 5.- GIRAFFE functional modules and their task summaries.

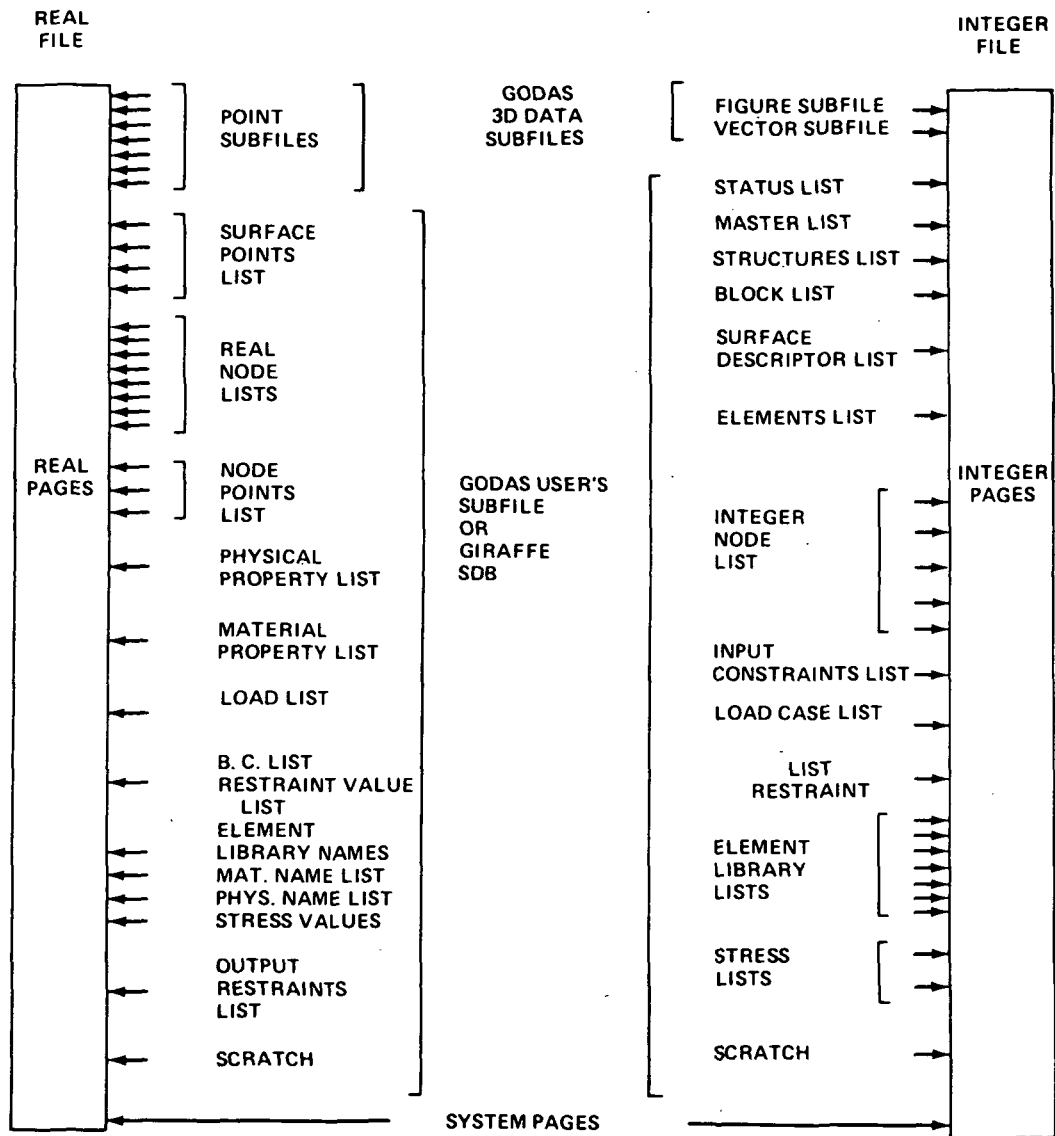


Figure 6.- Conceptual view of structural data base organization.

DATA BASE MANAGER
MACROS AND THEIR OPTIONS**

o MOVE (GDMOVER AND GDMOVI*)

$$\begin{aligned}\bar{A} &\rightarrow \bar{B} \\ -\bar{A} &\rightarrow \bar{B} \\ \bar{A} &\rightarrow \bar{B} \quad (\text{ELEMENT BY ELEMENT ABSOLUTE VALUE}) \\ \bar{A} * C &\rightarrow \bar{B} \\ \bar{A} + C &\rightarrow \bar{B}\end{aligned}$$

o NORM (GDNORMR)

$$\begin{aligned}\Sigma \bar{A} &\rightarrow C \\ \pi \bar{A} &\rightarrow C \\ \text{MAX } \bar{A} &\rightarrow C \\ \text{MIN } \bar{A} &\rightarrow C \\ \text{MAX } \bar{A} &\rightarrow C \\ \text{MIN } \bar{A} &\rightarrow C \\ \text{MIN (POS } A) &\rightarrow C \\ \text{MAX (NEG } A) &\rightarrow C\end{aligned}$$

o VECTOR (GDVECTR)

$$\begin{aligned}\bar{A} + \bar{B} &\rightarrow \bar{A} \\ \bar{A} - \bar{B} &\rightarrow \bar{A} \\ \bar{A} * \bar{B} &\rightarrow \bar{A} \\ \bar{A} / \bar{B} &\rightarrow \bar{A} \\ \bar{A} \cdot \bar{B} &\rightarrow C \quad (\text{DOT PRODUCT}) \\ \bar{A} - C \cdot \bar{B} &\rightarrow \bar{A} \quad (\text{ROW OPERATION})\end{aligned}$$

o EXTERNAL (GDEXTR AND GDEXTI*)

$$\begin{aligned}\bar{E} &\rightarrow \bar{A} \quad (\text{PUT ARRAY INTO DATA BASE}) \\ C &\rightarrow \bar{A} \quad (\text{BROADCAST CONSTANT}) \\ \bar{A} &\rightarrow \bar{E} \quad (\text{TAKE ARRAY OUT OF DATA BASE})\end{aligned}$$

WHERE \bar{E} IS AN EXTERNAL (USER DEFINED) ARRAY - MAY BE A PORTION OF PAGING BUFFER ALSO VIA RELEASE AND RESERVE CAPABILITIES

* FOR INTEGER MACROS

** OTHER MACROS PROVIDE FOR SPECIAL FUNCTIONS AS SPARSE MATRIX (VECTOR) OPERATIONS, MASKING, SELECTION, ETC.

Figure 7.- Data base manager macros.

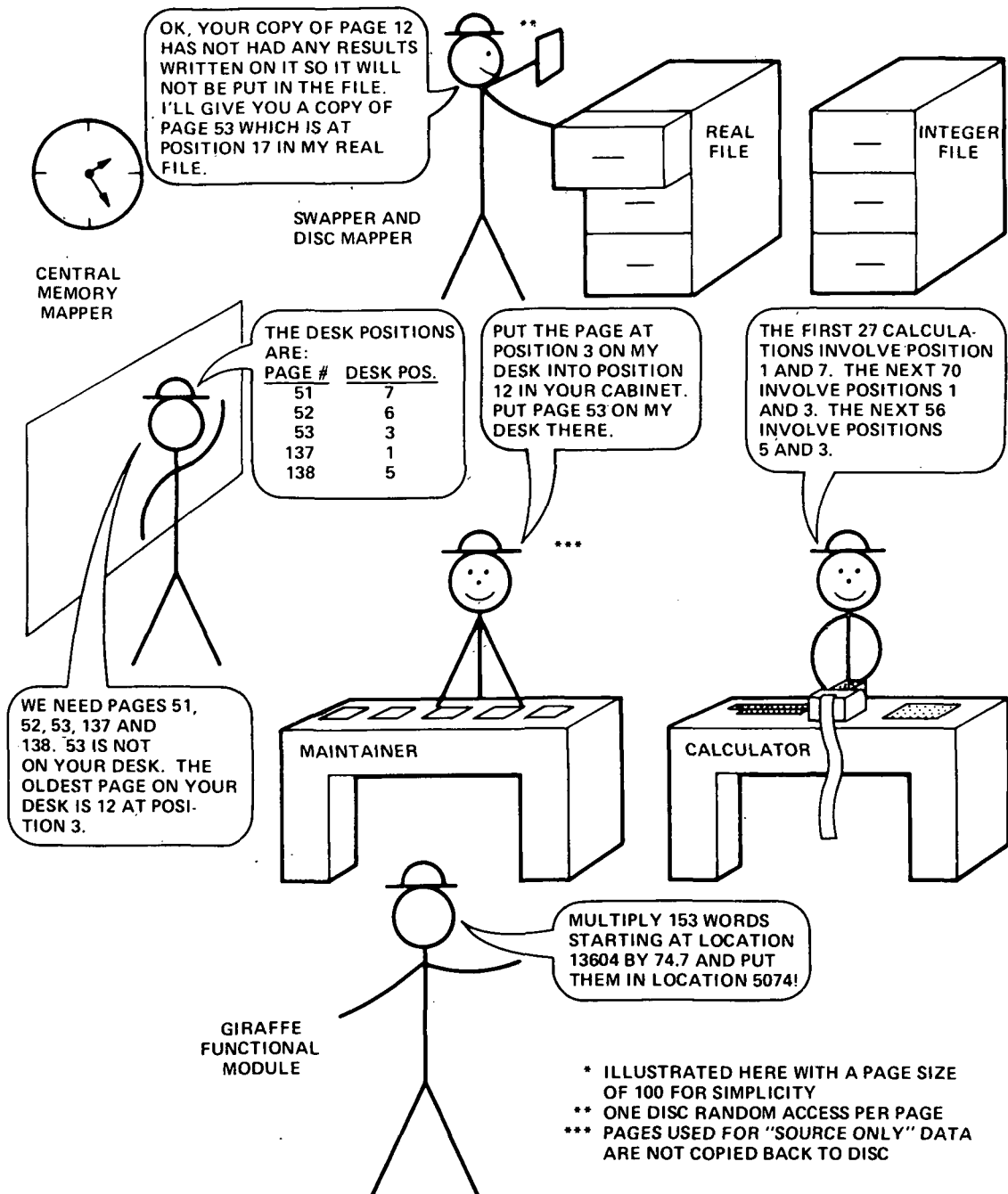


Figure 8.- Structural data base operation*.

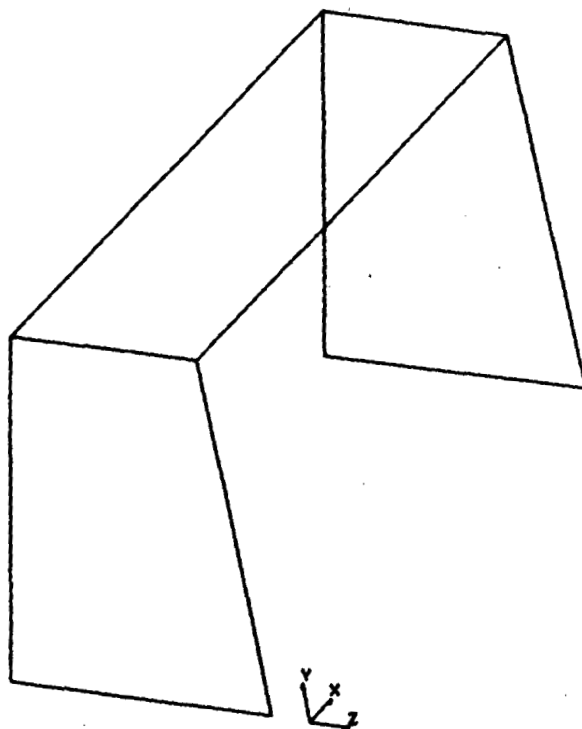


Figure 9.- Basic structure surface definition display
(CRT, batch or other input).

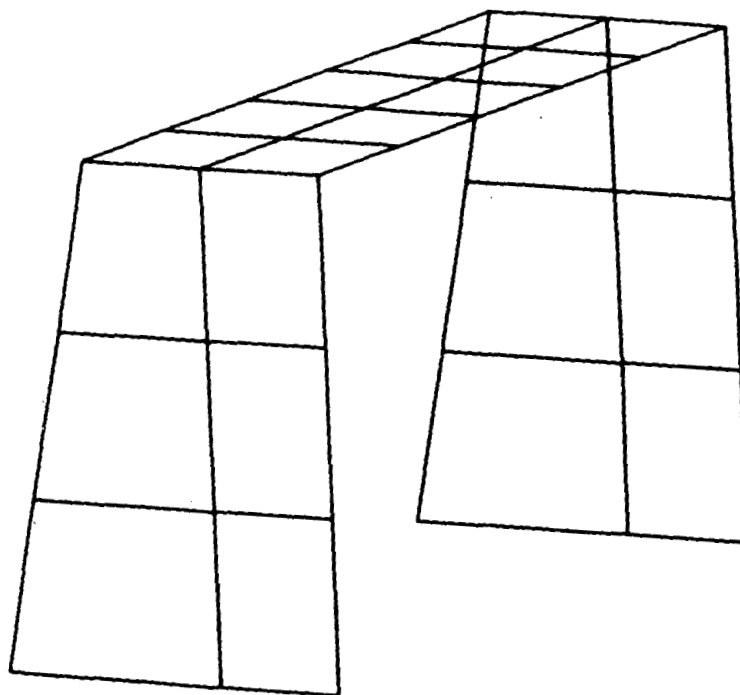


Figure 10.- Structure with finite element mesh.

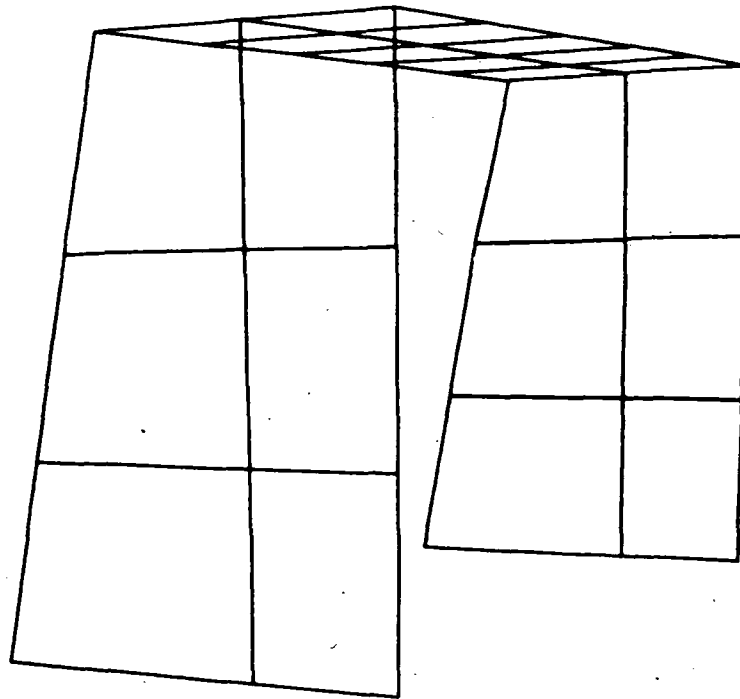


Figure 11.- Perspective view of finite element mesh.

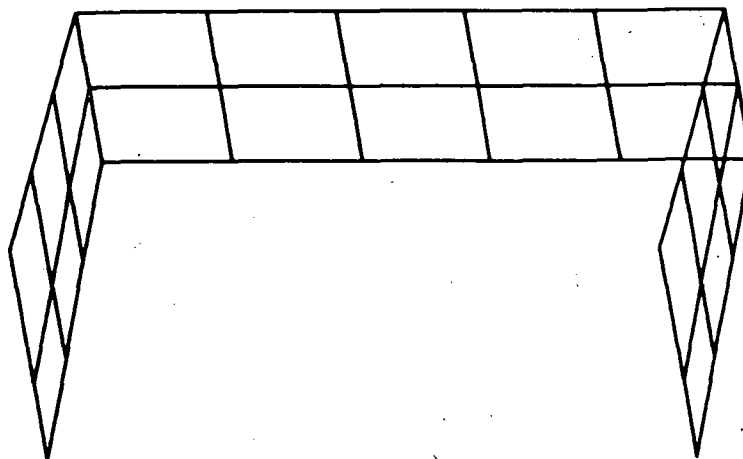


Figure 12.- Rotated view of structure.

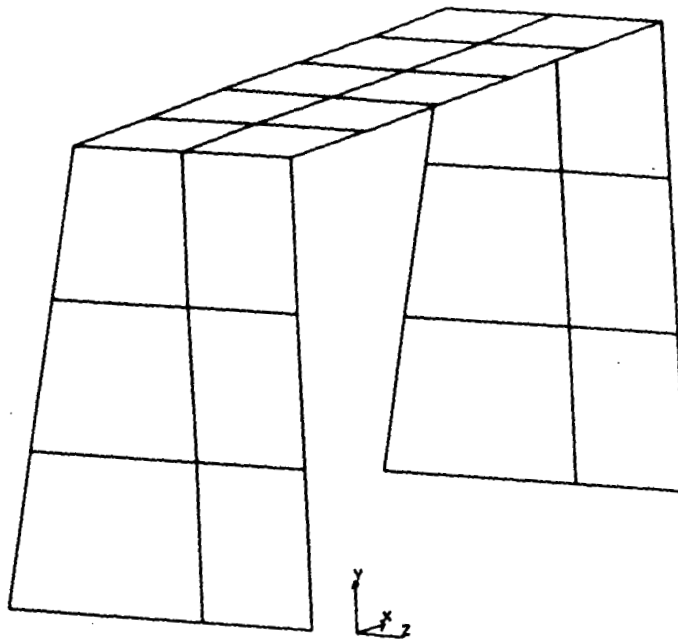


Figure 13.- Structure with hidden lines removed.

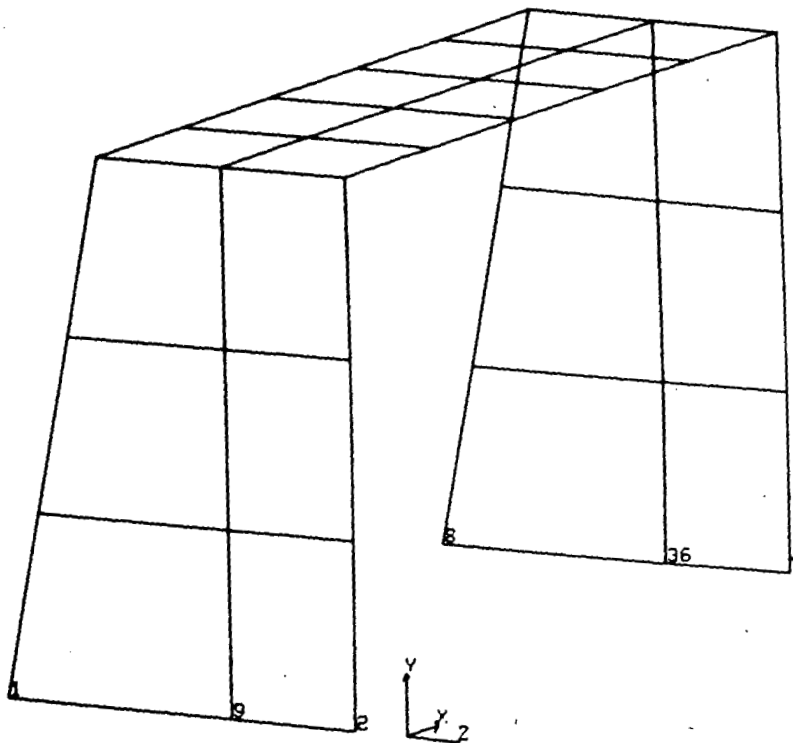


Figure 14.- Fully restrained node number display
(restrained in all six degrees of freedom).

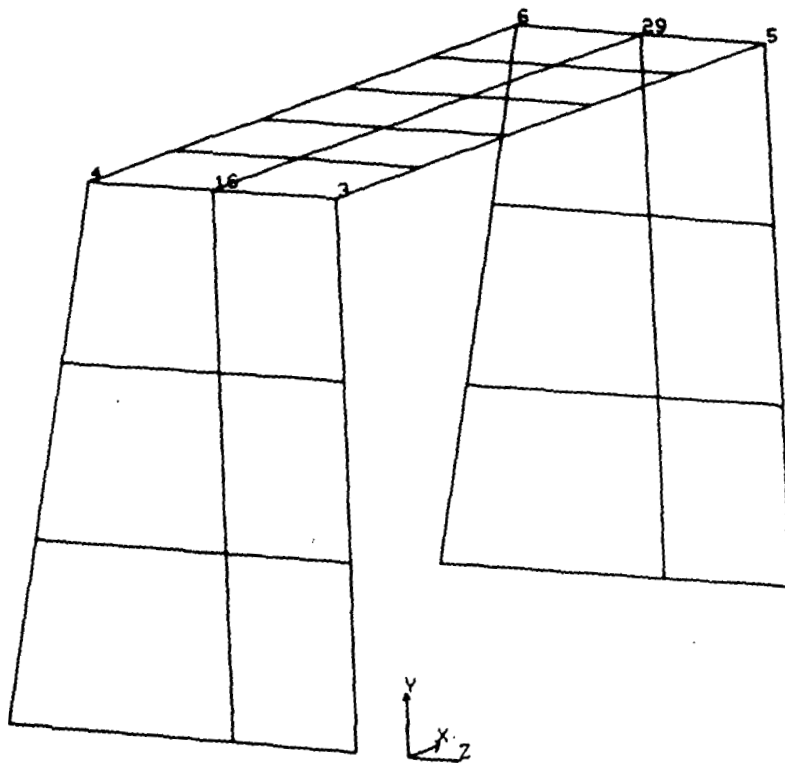


Figure 15.- Node number display for unrestrained nodes.

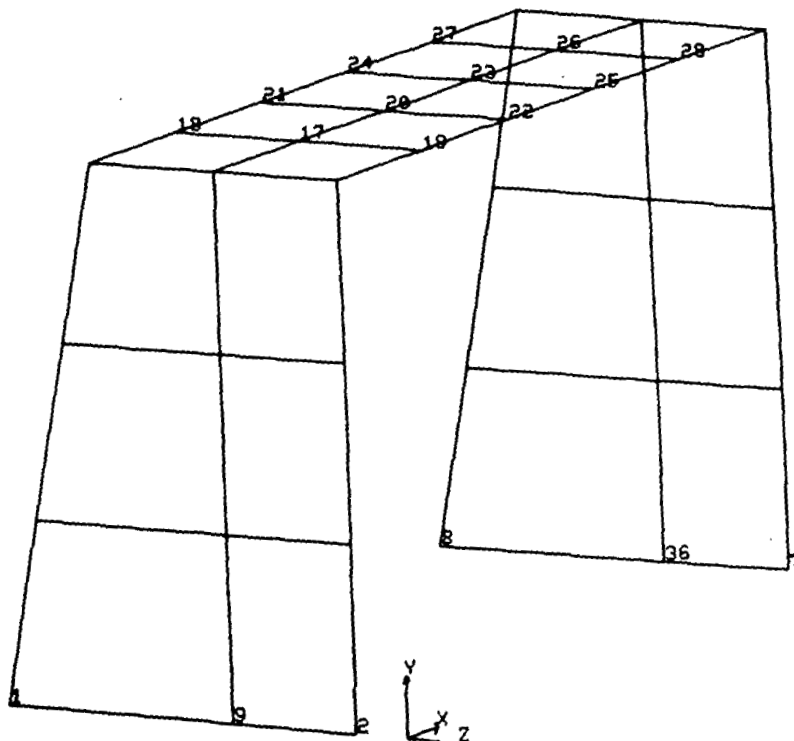


Figure 16.- Nodes restrained in RY degree of freedom.

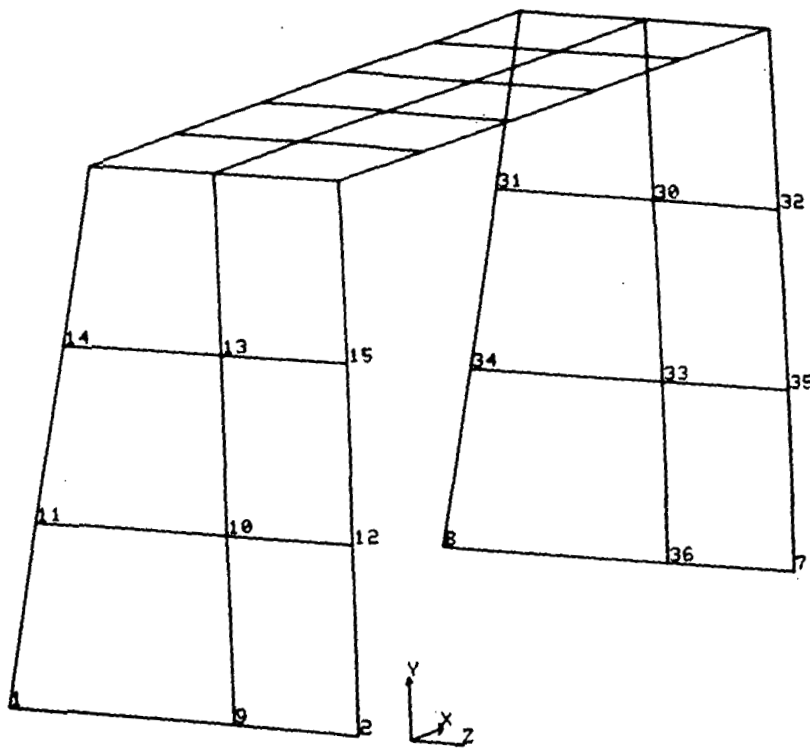


Figure 17.- Nodes restrained in RX degree of freedom.

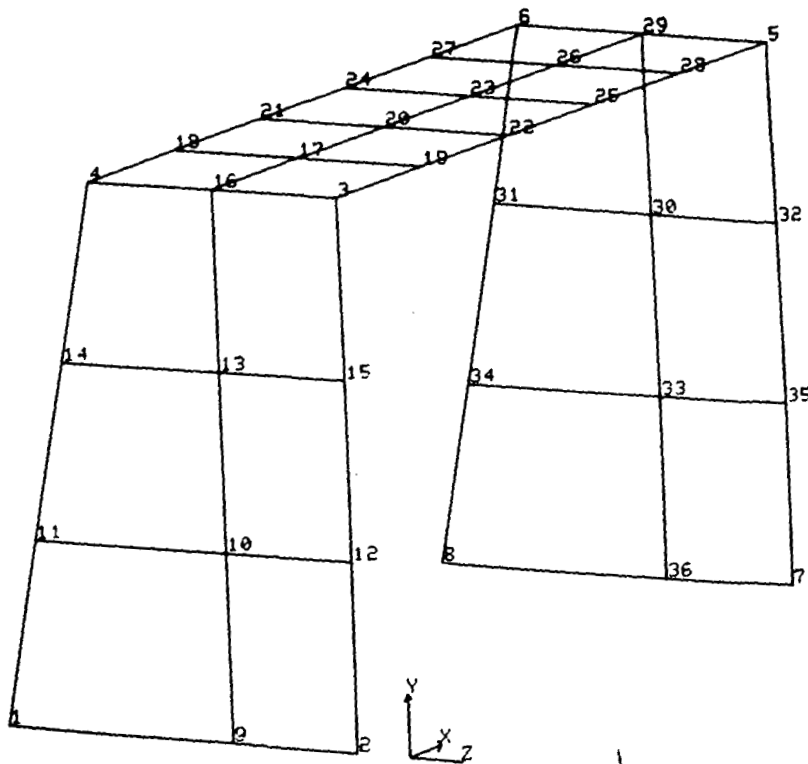


Figure 18.- Finite element model with user node numbering.

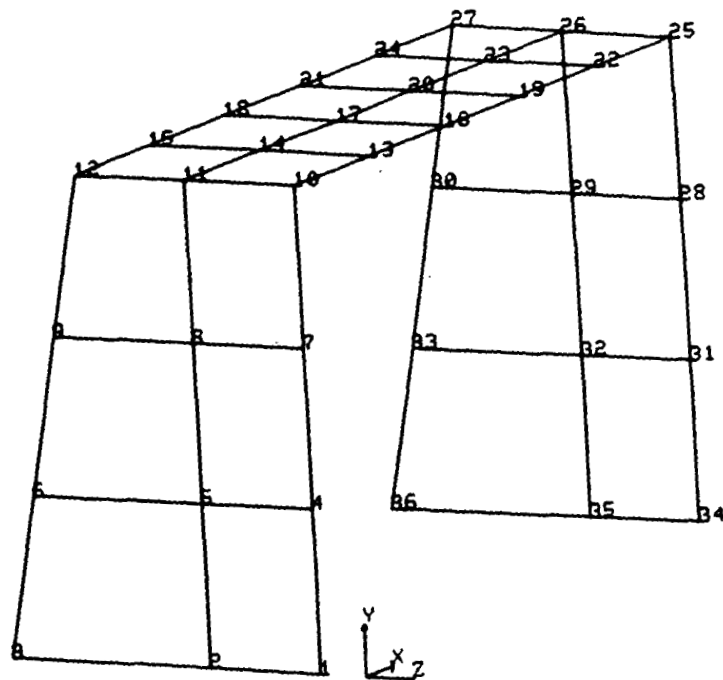


Figure 19.- Structure with optimized node numbering.

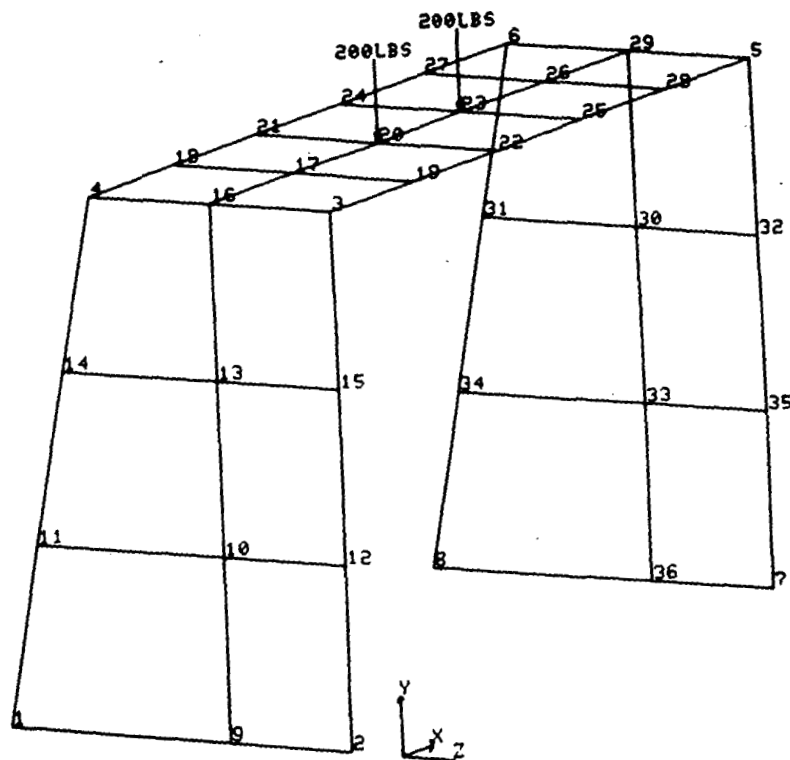


Figure 20.- Load specification with node number display option
(200 lbs = 90.7 kg).

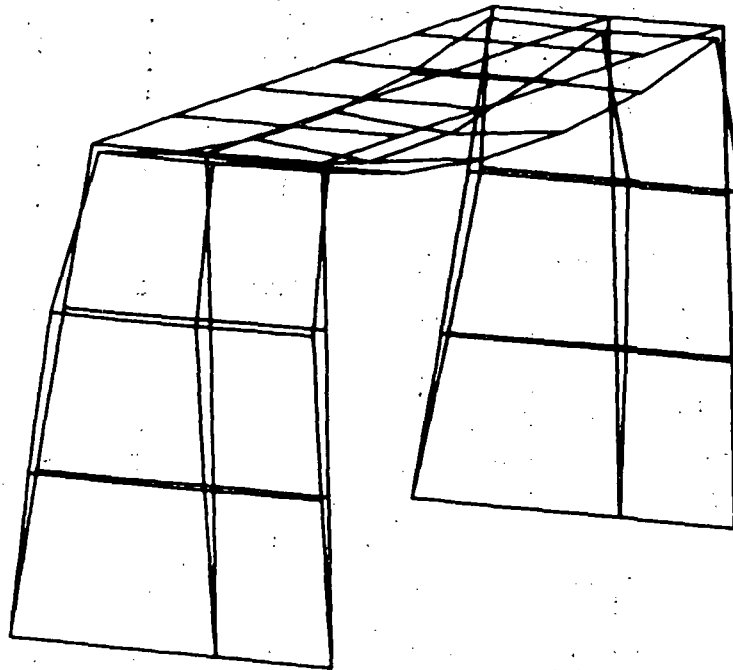


Figure 21.- Structure's deformation. Rotated view.
Scale factor = 3.0.

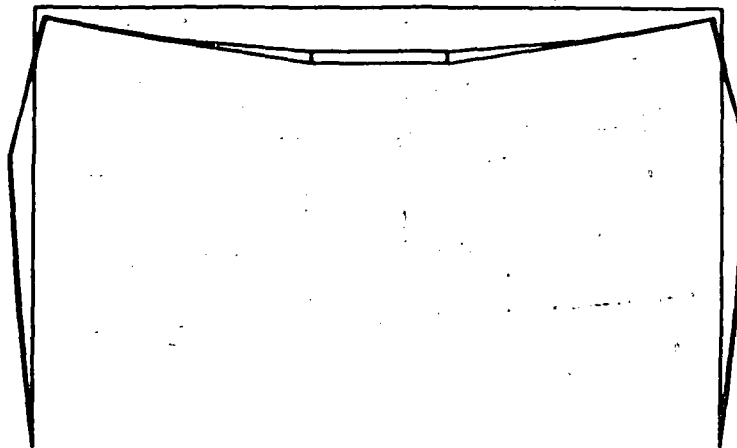


Figure 22.- Structure's deformation. Front view.
Scale factor = 3.0.

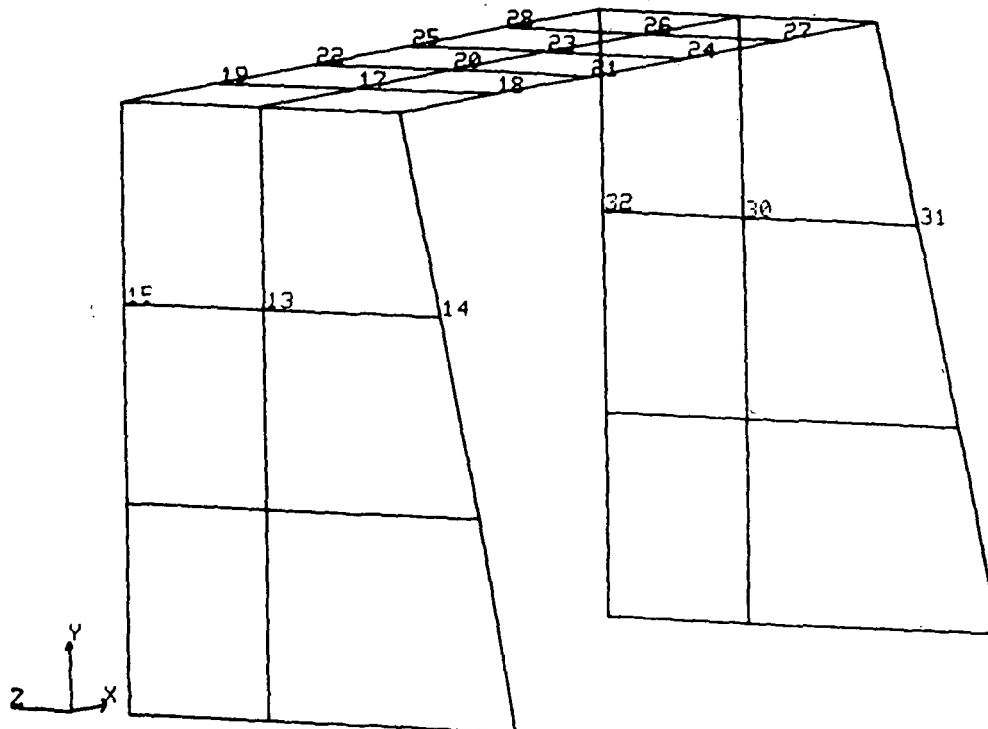


Figure 23.- Node point display for limit displacement (displacement above 0.025).

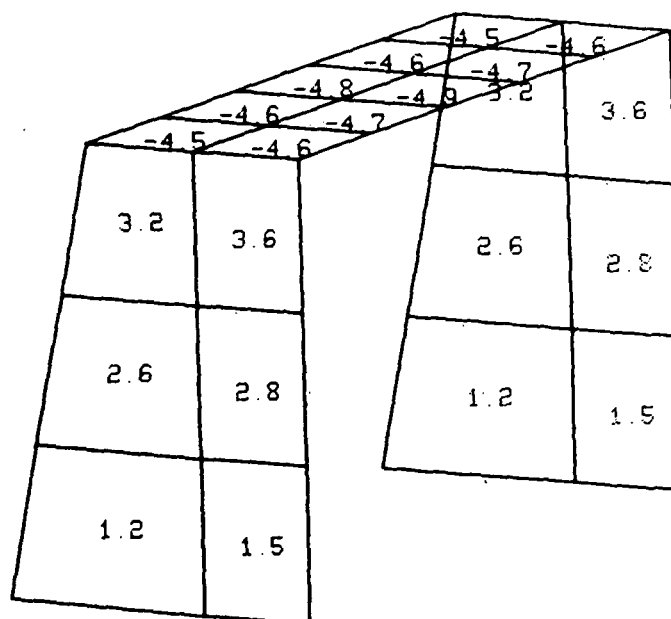


Figure 24.- Major principal stress (upper fiber, 6.85 MPa (1000 psi)) values on elements. (All may be displayed or only those above a specified value.)

-4.5	-4.6	-4.8	-4.6	-4.5
-4.6	-4.7	-4.9	-4.7	-4.6

Figure 25.- Rotated view of element stress plot, major principal stress (upper fiber, 6.85 MPa (1000 psi)).